

## COMPUTATIONAL ANALYSIS OF A SEDAN USING COMPUTATIONAL FLUID DYNAMICS SOFTWARE

SREERADH V M

*Research Scholar, Department of Mechanical Engineering, National Institute of Technology Calicut, Kerala, India*

### ABSTRACT

*This paper presents computational investigation of air flow over a sedan. This work presents a computational method to deduce the pressure and velocity beneath sedan. The study is done on two dimensional model of Honda S 2000 with different diffuser angles in creo parametric and analyzed in ANSYS FLUENT to obtain the surface velocity and pressure distribution. It is found that pressure and velocity are influenced by the angle of diffuser. The fluid used for this purpose is air.*

**KEYWORDS:** *Coefficient of Pressure, Computational Methods, Diffuser & Drag*

**Received:** Apr 12, 2018; **Accepted:** May 25, 2018; **Published:** Jun 14, 2018; **Paper Id.:** IJMPERDJUN2018104

### INTRODUCTION

A few years ago when oil crisis was not a problem, cars were mainly designed for high-speed manoeuvre, comfort and safety. The relevance of aerodynamic study of vehicles is the modification of car external to improve the car overall aerodynamic characteristics for better cruising conditions, greater stability of navigation, and lower energy consumption. This paper proposes an effective numerical model based on the Computational Fluid Dynamics (CFD) approach to obtain the flow structure around a passenger car with a diffuser. A diffuser helps to expand the high-velocity flow under the vehicle in the low-velocity ambient air, thereby reducing the flow separation. The flow field is determined by solving two-dimensional incompressible Navier-Stokes equations while the effects of turbulence are accounted for by the  $k$ - $E$  model.

### LITERATURE REVIEW

In the technical review presented by Satoshi Katoka et al. [1] the aerodynamics of the Lancer Evolution X is evaluated and also introduced the under floor air guide, a new aerodynamic device. The applied aerodynamic technology includes the nose shape like that of a shark, cooling, rear spoiler shape, etc. As a result the drag coefficient and lift coefficient values are less than that of the Lancer Evolution IX. Aerodynamics of road vehicles is studied in detail by Callister and George [2]. Influence of different diffuser angle on sedan's aerodynamic characteristics is a paper presented by Xingjun Hu et al. [3]. Here they adopted the method of CFD to study the aerodynamic characteristics of a simplified sedan with a different diffuser angle. The diffuser angle was set to 0°, 3°, 6°, 9.8°, and 12° respectively. Chin-Hsiung Tsai et al. [4] have studied the flow around Honda S 2000 in their paper Computational Aero-acoustic analysis of a passenger car with a rear spoiler. There are several other articles too discussing the basics of CFD and car aerodynamics [5-15], which will be helpful to understand the relevance of vehicle aerodynamics.

## MODEL PREPARATION

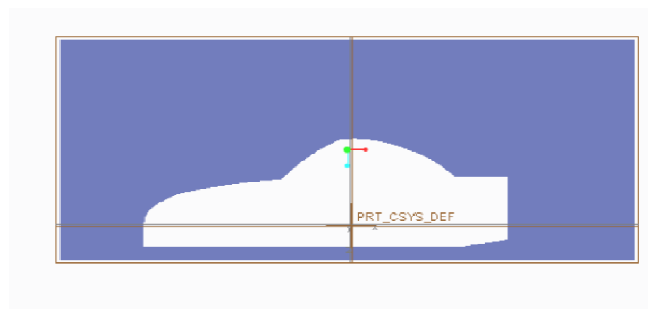
Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation is

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (1)$$

And the conservation of momentum equation is

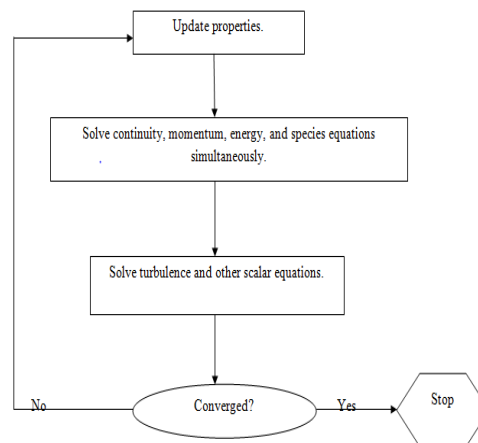
$$\frac{\partial(\rho u_i)}{\partial t} + u_j \frac{\partial(\rho u_i)}{\partial x_j} + \frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) - \rho F_i = 0 \quad (2)$$

The flow is assumed to be incompressible and steady. These conditions will simplify the equations. These equations along with the conservation of energy equation form a set of coupled, nonlinear partial differential equations. It is not possible to solve these equations analytically for most engineering problems. However, it is possible to obtain approximate computer-based solutions to the governing equations for a variety of engineering problems. This is the subject matter of Computational Fluid Dynamics. The first step in solving was to create a geometric model. The models were designed using Creo Parametric 2.0 as shown in Figure. 1.



**Figure 1: Geometric Model of Sedan in CREO**

The coupled solver solves the governing equations of continuity, momentum (where appropriate) energy, and species transport simultaneously (coupled together). Governing equations for additional scalars will be solved sequentially (i.e., segregated from one another and from the coupled set). Because the governing equations are non-linear and coupled, several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps and is explained below with the help of Figure. 2.



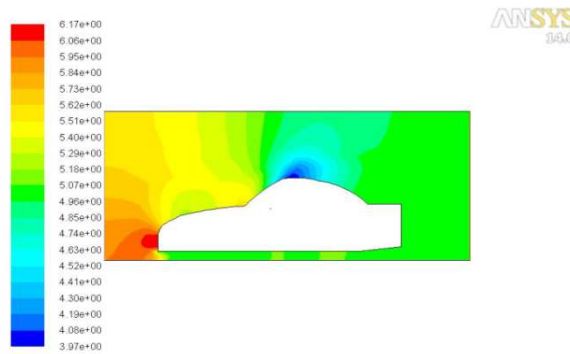
**Figure 2: Generalized Flowchart for Solution**

- Fluid properties are updated, based on the current solution, (if the calculation has just begun the fluid properties will be updated based on the initialized solution.)
- The continuity and momentum equations are solved simultaneously.
- Whereas the appropriate equations for scalars such as turbulence are solved using the previously updated values of the other variables.
- When the interface coupling is to be included the source terms in the appropriate continuous phase equations may be updated with a discrete phase trajectory calculation.
- A check for convergence set is made.
- It is here that the actual execution of the flow simulation begins.
- The convergence criteria are defined such that the solution will get converged when the value of all residues go below 0.001 (default value).
- Now the model is ready to be iterated.
- Thus the problem can be solved successfully.

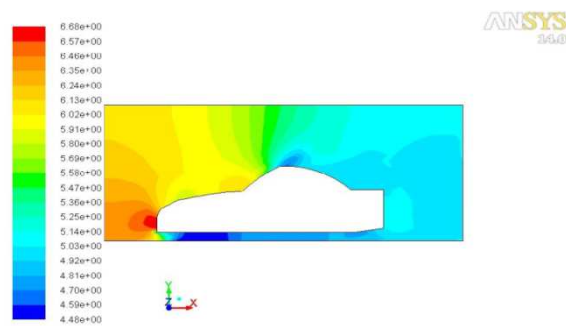
## RESULTS AND DISCUSSIONS

Current models were designed using CREO Parametric 2.0. Then it was imported to ANSYS Fluent. Three cases were selected for analysis with diffuser angles of  $7^\circ$ ,  $8^\circ$  and  $9.8^\circ$ . Car geometry was extruded out from a rectangular control volume. Flow through the control volume was then analyzed. The model was meshed for the purpose. The size function is a feature which enables us to set a gradient as desired, so as to vary the mesh density continuously throughout the area. More density was provided near the boundary of the car. Meshed pattern is created in all cases. Around 1200 nodes are created in each control volume. Fluid considered here is air with density  $1.2256 \text{ Kg/m}^3$  and viscosity  $1.714 \times 10^{-5} \text{ Kg/ms}$ . Inlet fluid velocity is given as  $100 \text{ m/s}$ . An outlet pressure boundary of 1 bar was given. Top and bottom of geometries was assigned wall boundaries. Continuity and energy equations were solved with these boundaries; ANSYS Fluent was the application used to solve domain. SIMPLE algorithm was used to solve the equations. A convergence was obtained after 1200 iterations. Velocity of fluid around the vehicle and coefficient of pressure (COP) was obtained as results. The Figures

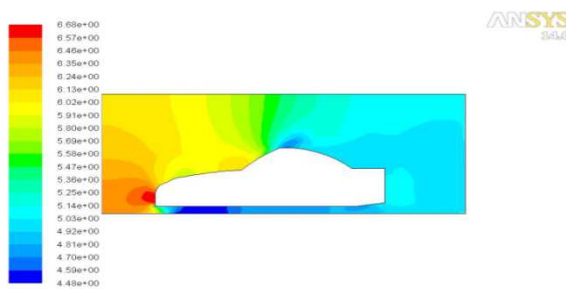
3-8 presents the contours.



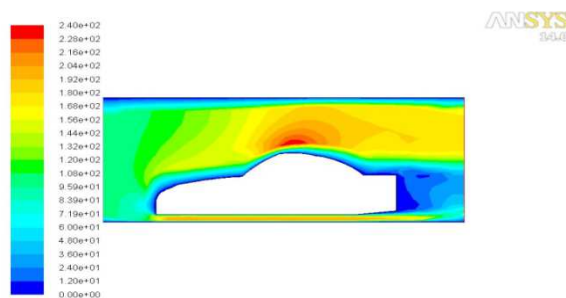
**Figure3: COP for 7° Diffuser Angle (Case 1).**



**Figure4: COP for 8° Diffuser Angle (Case 2).**



**Figure5: COP for 9.8° Diffuser Angle (Case 3).**



**Figure 6: Velocities for 7° Diffuser Angle (Case 1).**

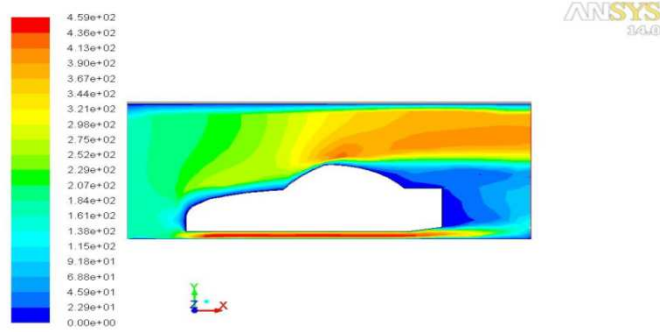


Figure 7: Velocities for 8° Diffuser Angle (Case 2)

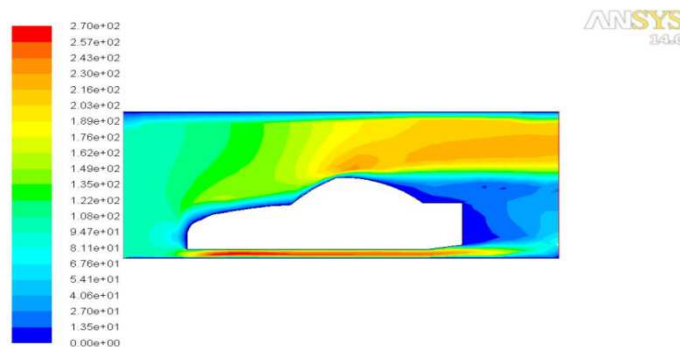


Figure 8: Velocities for 9.8° diffuser angle (Case 3)

## CONCLUSIONS

Diffuser tries to increase the velocity beneath the vehicle and merge the flow with downstream. This creates a low-pressure region underneath and the pressure differences between the top and bottom of the vehicle will increase. This will press the car more to the road or it reduces the lift on the car and increases the traction. We can read the values of the coefficient of pressure and velocities from the contours. It can be observed that pressure is almost similar up to the middle part. On reaching the diffuser it varies in all cases. Same is the case with velocity. But velocity is more (459m/s for 8° and 270m/s for 9.8°) air pressure is low beneath the sedan in the 8° diffusers. This will force the vehicle more to the ground and thereby be giving better traction and fuel efficiency. So it's better to use an 8° diffuser angle than 9.8° or 7° diffuser angle.

## REFERENCES

1. Satoshi Kataoka, Norimasa Hashimoto, Masahiro Yoshida, Tomio Kimura, Naoki Hamamoto, *Aerodynamics of Lancer Evolution X*, Mitsubishi motors, Technical review, 2008.
2. J. R. Callister, A. R. George, *Aerodynamics of Road Vehicles*, in wind noise: W. H. Hucho (Ed.), SAE International, Warrendale, PA, 1998.
3. Xingjun Hu, Rui Zhang, Jian Ye, Xu Yan, Zhiming Zhao. *Influence of Different Diffuser Angle on Sedan's Aerodynamic Characteristics*. International Conference on Physics Science and Technology, 2011.
4. Chien-HsiungTsaia, *Computational aero-acoustic analysis of a passenger car with a rear spoiler*, Applied Mathematical Modelling, 2009.
5. H. Taeyoung, V. Sumantran, C. Harris, T. Kuzmanov, M. Huebler, T. Zak, *Flow-field simulations of three simplified vehicle shapes and comparisons with experimental measurements*, SAE Transactions 106 (1996) 820–835.
6. J. H. ferziger, M. Peric, *Computational methods for fluid dynamics*, Springer publications.

7. J. P. Howell. *The Influence of Ground Simulation on the Aerodynamic of Simple Car Shapes with an Under floor Diffuser*, Conference on Vehicle Aerodynamics. Royal Aerodynamic Society, 1994.
8. R. Cooper Kevin, T. bertenyi, G. Dutil, J. sym, G. Sovran. *The aerodynamic performance of under body diffuser*, SAE Technical Paper 980030, 1998.
9. Shah, T., & Tailor, M. *A Numerical Solution Of Laminar Flow In Porus Media With Triangular Duct By Finite Difference Method With Matlab*.
10. Ye Hui, *A Parametric Study on the Diffuser and Ground Clearance of a Simplified Car Model using CFD*. Jilin university, 2006.
11. Reddy, P. R., & Saikiran, M. *Aerodynamic Analysis of Return Channel Vanes in Centrifugal Compressors*. *International Journal of Mechanical Engineering (IJME)*, 5(1), 73-82.
12. J. Cederlund, and J. Vikström, *The Aerodynamic Influence of Rim Design on a Sports Car and its Interaction with the Wing and Diffuser Flow*. M.Sc. Thesis, Chalmers University of Technology, 2010.
13. Moorthy, C. V., Srinivas, V., Prasad, V. V. S. H., & Vanaja, T. (2017). *Computational Analysis Of A CD Nozzle With „SED” For A Rocket Air Ejector In Space Applications*. *International Journal of Mechanical and Production Engineering Research and Development*, 7(1), 53-60.
14. Lasse Christoffersen, David Söderblom, Lennart Lofdah, *Wing-Diffuser Interaction on a Sports Car*. SAE Technical Paper, 2011-01-1433, 2011.
15. Raut, R. V., & Chaudhari, K. M. *Cfd Analysis Of Solar Heater Water Pipe With Different Inclination*.
16. Fu Limin. *Automobile Aerodynamics*. Beijing: China Machinery Press, 2006.
17. Hu Xingjun, Qin Peng, GuoPeng, An Yang. *Effect of Turbulence Parameters on Numerical Simulation of Complex automotive External Flow Field*. *Applied Mechanics and Materials*, 2011, vols. 52-54, pp. 1062-1067.
18. Hu Xing-Jun, YangBo, Wang Jing-Yu, Li Ting. *Research on influences of rear-view mirror on aerodynamic drag characteristics of truck*. *Journal of Hunan University Natural Sciences*, 2010, v37, n SUPPL. 1, pp. 65-69.
19. Wang Fujun. *Computational Fluid Dynamics*. Beijing: Tsinghua University Press, 2004.

